

Study on Hydrodynamic Performance of Podded Propulsion in Viscous Flow

Shen Xingrong, Feng Xuemei, Cai Rongquan, Cai Yuejin

Marine Design and Research Institute of China (MARIC), Shanghai, China

ABSTRACT

The improvement and the development of podded propulsion hydrodynamic performance research by the model test and the numerical simulation are reviewed. A numerical model of podded propulsor has been established, and the moving mesh is successfully applied to simulate the influence of running propeller.

The open water performance characteristic of podded propulsor in unsteady viscous flow is numerically predicted. The computed results are compared with the model test results, the maximal difference of K_T is about 2.9%; K_Q is about 0.8%. It shows that the prediction technique developed in this paper is feasible. Further more some significant hydrodynamic phenomena of podded propulsor are focused and illustrated, including the pressure distribution on podded propulsor unit, the distribution of velocity vector around propeller plane and the characteristics of flow around the podded propulsor. The interaction between propeller and pod housing is displayed.

Keywords

podded propulsor; hydrodynamic performance; moving mesh; viscous flow; unsteady flow

1 INTRODUCTION

In the last decade, the podded propulsion has been rapidly becoming more and more important as main propulsive unit for a wide class of ships, including navy and commercial vessels. This has brought the significant problems of hydrodynamic performance predictions for podded propulsors. A lot of research work in this field has been done in the world wide by experiments or numerical simulations. The 22nd ITTC suggested to draft a propulsion prediction procedure for ship with podded propulsor (ITTC 1999). The 23rd ITTC has established a Specialist Committee with the task developing and improving the procedures for podded propulsor tests and extrapolation to full-scale (ITTC 2002) and the 24th ITTC has recommended the procedures for carrying out podded propulsor model tests, cavitation experiments and extrapolation to full-scale (ITTC 2005). Many experimental research projects have been collaborated among different basins. Various podded models, testing methods and extrapolation procedures have been developed for predicting the full-scale performance (Veikonheimo 2006). On the other hand, the

technique of numerical simulations for podded propulsor hydrodynamic performance has been developed quickly, including potential methods, viscous methods and coupling method between potential and viscous methods (Krasilnikov et al 2007), (Krasilnikov et al 2005). Potential methods which are based on lifting surface and panel method are very efficient and convenient (Ma et al 2004). Generally the panel method is applied to simulate the pod housing and the vortex lattice method is applied to simulate the propeller. Iterations are carried out between the two methods till convergence is obtained and the complete of the interaction between the propeller and the pod housing is captured (Islam 2004), (Gupta 2004), (Mishra 2005). However, the potential methods require some empirical corrections for the effect of viscosity. This brings some uncertain aspects about potential methods. Obviously, it is a good idea to solve this problem by using RANS or Navier-Stokes solvers to simulate the podded propulsor unit in viscous turbulent flow (Sanchez Caja et al 2004), (Chicherin et al 2004). The practical calculations demonstrate the viscous methods are quite sensitive to meshing, turbulence model and treatment of the boundary conditions. The principal difficulty of viscous methods is the technology for modeling of the rotating part of podded propulsor, i.e. the rotating propeller. It is a hard work to solve this principal problem. The complete viscous methods are also the most time and resource consuming. So a compromised solution is presented by so-called coupled viscous-potential methods where pod housing is simulated in viscous flow while propeller is subject to potential calculation. In even simpler manner, some scholars applied an actuator disk scheme to simulate the effect of propeller in viscous solution, and this method ignores the fact that velocities in the propeller plane are unsteady and non-uniform.

In the present paper a viscous method with actual geometrical propeller model is applied to predict the hydrodynamic performance of podded propulsor in open water. The computation results are compared with the experimental data. By the numerical simulation, some significant hydrodynamic phenomena of podded propulsor are illustrated, including the pressure distribution on podded propulsor unit, the distribution of velocity vector around propeller plane and the characteristics of flow around the podded propulsor. The meshing was performed in the commercial CFD software FLUENT's pre-processing program GAMBIT. The

solution of viscous flow around podded propulsor was done by use of FLUENT.

2 Modeling

2.1 Geometric Model

The model employed by the computation case was a pulling-type podded propulsor, including 5 blades propeller, hub, hub cap, strut, pod housing and tail fin. Figure 1 shows the sketch of the podded propulsor. Table 1 gives the main parameters of the podded propulsor.

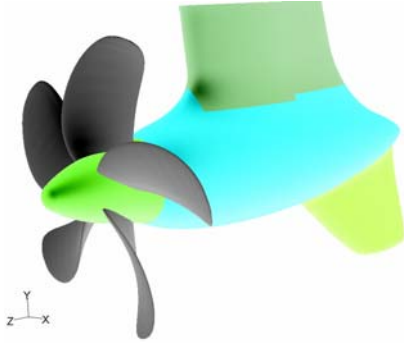


Figure 1 Sketch of the podded propulsor

Table 1 Main Parameters of Podded Propulsor (Model Scale)

| | Symbol | Unit | Data |
|-------------------------|--------------|------|--------|
| Number of blades | N | [-] | 5 |
| Diameter | D | [m] | 0.200 |
| Mean pitch ratio | P_{mean}/D | [-] | 1.0412 |
| Area ratio | A_E/A_O | [-] | 0.614 |
| Skew angle | θ_s | [°] | 22.09 |
| Hub diameter ratio | dh/D | [-] | 0.24 |
| Length of pod housing | L_p | [m] | 0.2759 |
| Diameter of pod housing | d_p | [m] | 0.0867 |

This study was carried out at model scale, and the simulated working conditions were same as the ones of the model open water experimental environment in order to compare the results between them.

2.2 Computational domain

The coordinate system origin located at the point where the propeller reference line intersected with the rotation axis of propeller. The z-axis paralleled to the propeller axis and upstream pointing. Computational domain was defined with dimensions $-2.7L_p < z < 1.3L_p$ in main flow direction, $-1L_p < x < 1L_p$ in lateral direction and $-1.5L_p < y < 1.5D$ in vertical direction. Figure 2 shows the sketch of computational domain and coordinate system. The dimension range of this computational domain and other numerical parameters used in this case are decided based on author's experience from some other series study in this area(Feng et al 2006).

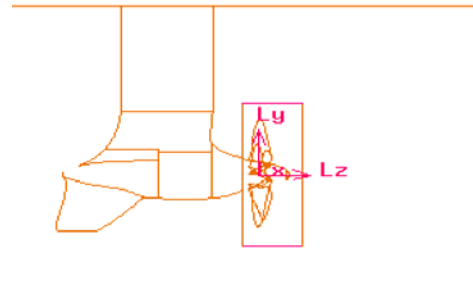


Figure 2 Sketch of computational domain and coordinate system

2.3 Grid Generation

The moving mesh is applied to simulate the rotating effect of the actual geometric propeller. To obtain an accurate flowing field solution and stable convergence, a good mesh quality is required and unsteady time-step iterations are performed. The rotating part in the mesh is a cylindrical block, just big as entirely enclosing the blades. Sliding surface was set between the rotating part and the fixed part. The cell number of rotating part is about 0.68 million, and the number of fixed part is about 0.8 million. So the total cell number is about 1.5 million. The grid of rotating part and fixed part are shown in Figure 3.

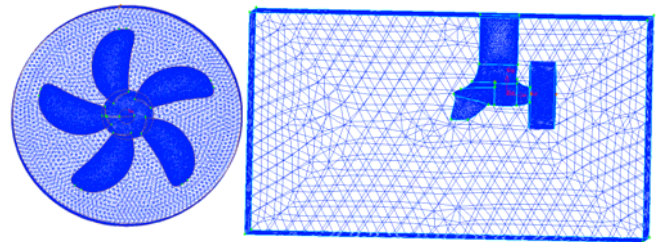


Figure 3 Grid of the computing domain(Left is rotating part, Right is fixed part)

3 Numerical Modeling

3.1 Solve Strategy

A 3D unsteady time-step iterations, pressure based and segregated implicit solver was applied. The viscous solvers are very sensitive to turbulence model and discretization scheme. In this work, the realizable k- ϵ turbulence model was used; PISO algorithm was selected for pressure-velocity coupling and PRESTO was used as pressure discretization scheme; second-order upwind was used for the discretization scheme of momentum, turbulence kinetic energy and turbulence dissipation rate. In this case, the result shows that the convergence, reliability and stability of this solve strategy are quite satisfied.

3.2 Initial and Boundary Conditions

In this case, the rotating of propeller was set to 15rps as the one in model test. The range of advance coefficient was considered, i.e., $0.4 \leq J \leq 0.90$. The value of J was varied by changing V_a , while propeller rotation was kept constant, where V_a was propeller advance velocity in [m/s]. The Reynolds Number of blades was about 3.6×10^5 ; the Reynolds Number of pod housing was about 2.0×10^6 .

Boundary conditions were set to simulate the flow around the running podded propulsor in open water: on the inlet boundary, velocity components of uniform stream with the given inflow speed were imposed; on the outlet boundary, the static pressure was set to a constant value; on the sides and bottom boundary, the velocity inlet boundary condition was imposed; on the blades, hub, hub cap, pod housing, strut and fin surface, the no slip wall condition was imposed.

4 Results

Unsteady flow simulation in open water condition was carried out for the podded propulsor. The time step size was set to 0.0003333 [sec.], which corresponded to the rotation angle of 1.8 degree per time step. This propulsor had been studied experimentally in towing tank(Heinke et al 2006),(B.Günther 2006). Some results are compared with that of CFD and are presented below.

4.1 Open Water Characteristics

The calculated podded propulsor unit characteristics are compared with the measured values in Figure 4. In the figure the red dashed lines show present calculated data and the blue lines show the experimental data for comparison. We can see the characteristics of podded propulsor are predicted with satisfactory accuracy by numerical simulation, the difference in the prediction of K_T is less than 2.9% over the whole range of the advance coefficient J . Especially the difference of K_Q is less than that of K_T . The maximal difference of K_Q is about 0.8% for the podded propulsor. At lower advance coefficient J values, thrust and torque of podded propulsor were slightly underpredicted from the experimental data; at higher J value K_T and K_Q were slightly overpredicted compared with the measurements. In the whole range of J , the efficiency is lower than model test results. This tendency is similar to the results from the potential method. In author's opinion, one of the reason for that lies in the turbulence intensity difference between model test and numerical simulation. This leads to increasing or decreasing the resistance of pod housing compared to measurements.

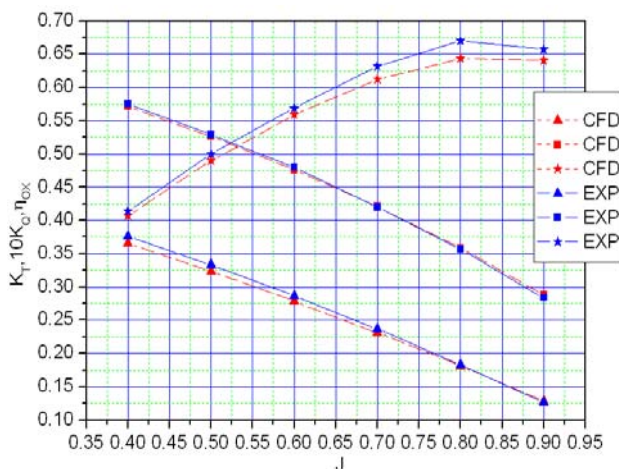


Figure 4 Computed and measured performances of podded propulsor in open water

4.2 Force on Pod Components

Figure 5 shows the force curves of pod components, i.e. the blades, hub, hub cap, pod housing, strut and fin. As a result, one can see the blades force is a big magnitude, while forces on pod housing, strut, fin, hub and hub cap are all small values. Among the small components, force on pod housing plays a principal role. In the range of studied advance coefficient, the drag of pod housing is about 3~6% of thrust of blades. It is observed an interesting phenomenon that at lower advance coefficient, hub and fin generate thrust, which contributes to the unit total thrust; while at higher J , they become to drag, which plays negative role to total thrust. In the whole range of advance coefficient, pod housing, strut and hub cap generate resistance, which decrease the efficiency of podded propulsor unit.

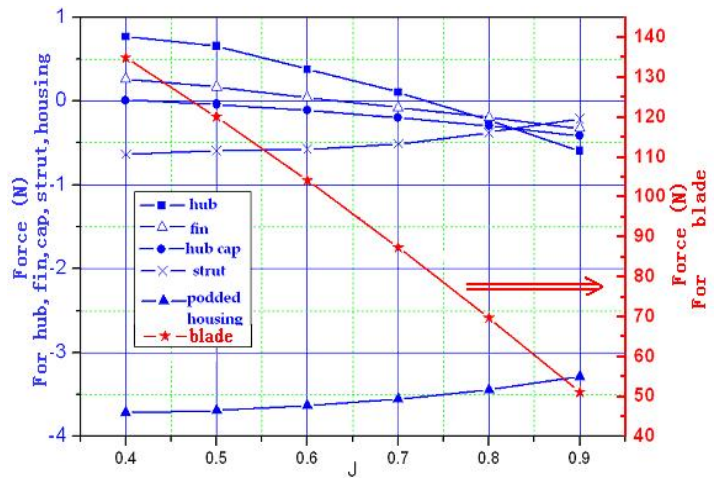


Figure 5 Force on Pod Propulsor Components

4.3 Flow Field around Podded Propulsor in Open Water Condition

Podded propulsor hydrodynamic performance is largely characterized by the interacting between the rotating propeller and fixed components(pod housing, strut, fin). And the flow field is intrinsical unsteady, regardless of straight inflow or of the oblique flow. The unsteady character can be obviously seen from Figure 6, the curves of blades force and podded propulsor unit force according to time series. The period of force fluctuation is about 0.013 seconds, which is consistent with the frequency of blade.

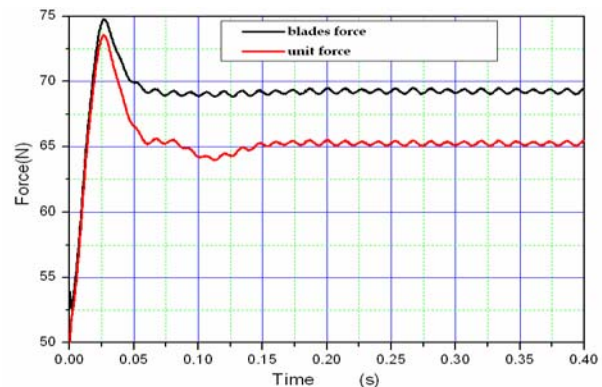


Figure 6 Time Series of Blades Force and Unit Force(J=0.8)

So, obviously, the pressure on blades and pod housing is fluctuating, and the velocity vector is varying in time domain. These hydrodynamic fluctuations are sources of noise, vibrations, cavitation and risk of erosion. This unsteady loading on podded propulsor bearings may be responsible of severe structural problems such as bearings failure and fatigue etc. The computed results are very useful for optimizing the geometry of podded propulsor so as to reduce the damage of fluctuating.

Figure 7 shows the pressure distribution on every component of podded propulsor. The lowest pressure appears at the leading edge near the propeller tips, especially when blade is at the position of 12 o'clock. The small low pressure area occurs at the suction side of the strut and fin, which locate in the downstream of the running propeller. Also we can see the pressure gradient is very large at the region from rotating hub surface to fixed pod housing.

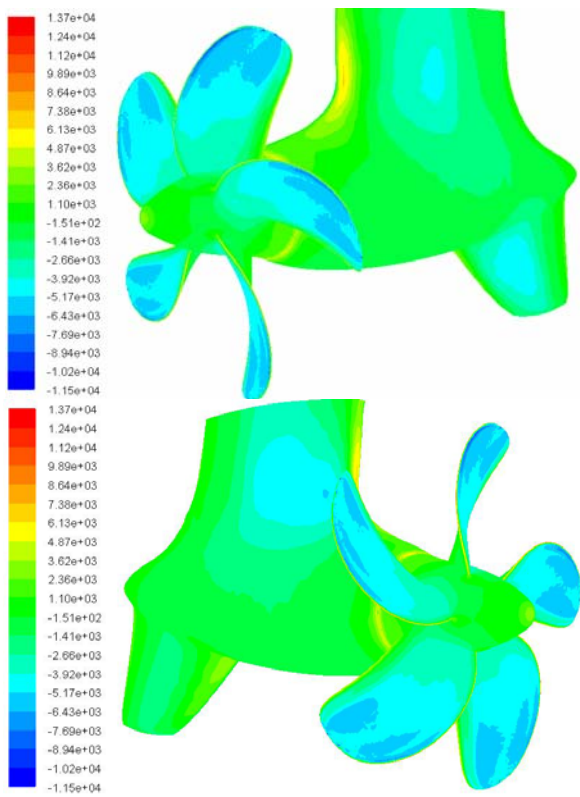


Figure 7 Pressure Distribution on the Podded Propulsor (Up: port view; Low: starboard view . J=0.8)

Also we can see the influence of running propeller on the velocity vector. Figure 8 shows the velocity vector in the planes which parallele propeller plane. In the upstream plane, we can see the suction effect clearly on the blades suction side, especially in the range from 0.5radii to 0.8radii. The flow shows three dimension influence intensively. The complex flow phenomena can be found on the tip range of the blades. In the downstream plane, the tip vortex has been distinctly shown (the white circle area), and the velocity distribution along the trailing edge in this plane is very complexity.

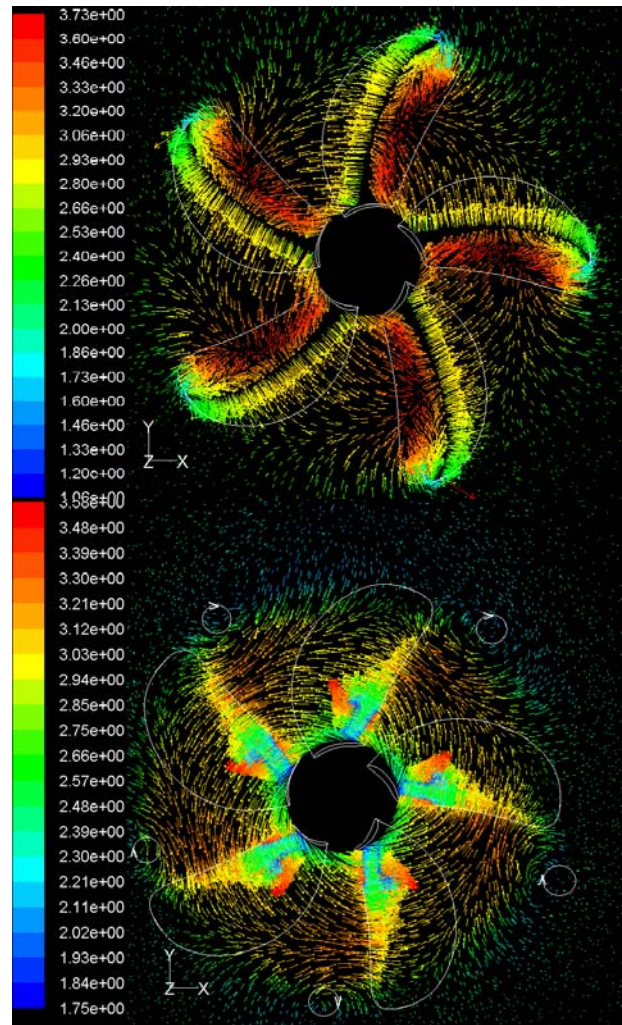


Figure 8 Vector Plot in the Plane Paralleled to Propeller Plane (Up: upstream 0.02D to propeller center plane ; Low: downstream 0.05D to propeller center plane. J=0.8)

Figure 9 shows streamlines on the pod housing, strut and fin surface. The streamlines run smoothly on the surface. It is apparent that the flow separation occurs at the end of pod housing and may generate the trailing vortex.

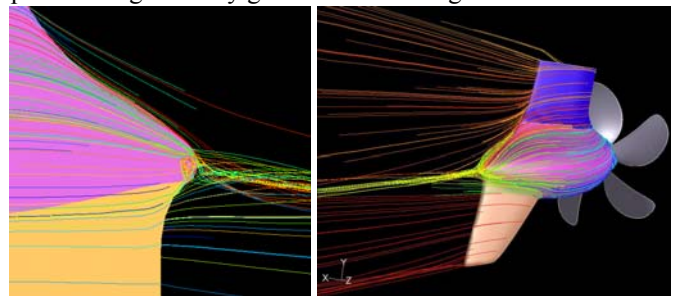


Figure 9 Streamlines on the Surface of podded propulsor (J=0.8)

5 Conclusions

The incompressible viscous flow around podded propulsor has been simulated by solving RANS equations with the RKE turbulence model and the open water performance characteristic of podded propulsor in unsteady flow has been predicted. The moving mesh is successfully applied to simulate the mutual influence of

running propeller and fixed components (i.e. hub, hub cap, pod housing, strut and fin). The computed results are well agreement with that got from experiments. It shows that the prediction technique developed in this paper is feasible. Further more, some details of flow field can be got from the numerical simulation and these information will be very useful and helpful to propeller designers. One can make some efforts to optimize the geometric of blades, strut, fins, hub or pod housing according to these flow information.

6 Acknowledgement

The authors would like to thank Dr Wang ping, Dr Fan Sheming and Mr. Gong Yinpu for supporting the present study.

REFERENCES

- Apurva Gupta(2004), Numerical Prediction of Flows Around Podded Propulsors, Presented to the Faculty of the Graduate School of The University of Texas at Austin in Partial Fulfillment of the Requirements for the Degree of Master of Science in Engineering, USA.
- Antonio Sanchez-Caja, Jaakko V. Pylkkänen(2004), On the Hydrodynamic Design of Podded Propulsors for Fast Commercial Vessels, Proceedings of The First International Conference on Technological Advances in Podded Propulsion, Newcastle, UK.
- Bikash Mishra(2005), Prediction of Performance of Podded Propulsors via Coupling of a Vortex-Lattice Method with an Euler or a RANS solver, Presented to the Faculty of the Graduate School of The University of Texas at Austin in Partial Fulfillment of the Requirements for the Degree of Master of Science in Engineering, USA.
- B.Günther(2006), Resistance Test, Propulsion Test and Wake Measurement, Report of SVAtch GmbH Potsdam.
- Feng Xuemei, Cai Rongquan, et al(2006), The Numerical Simulation of the Flow Field around a Propelled Hull Including Free Surface Influence. Selected Papers of CSNAME (Supplement of Shipbuilding of China), Vol.17.
- H.-J.Heinke, H.Richter(2006), Cavitation Test and Pressure Pulse Measurements with The Azipull-thrusters, Report of SVAtch GmbH Potsdam.
- I.A. Chicherin, M.P. Lobatchev, A.V. Pustoshny, A. Sanchez-Caja(2004), On a Propulsion Prediction Procedure for Ships with Podded Propulsors Using RANS-Code Analysis, Proceedings of The First International Conference on Technological Advances in Podded Propulsion, Newcastle, UK.
- Ma Cheng, Qian Zhengfang, Yang Chenjun, Zhang Xu, Du Du(2004), Research On Hydrodynamic Computation Model of Pod Propulsion, Proceedings of The First International Conference on Technological Advances in Podded Propulsion, Newcastle, UK.
- Mohammed Fakhru Islam(2004), Numerical Investigation on Effects of Hub Taper Angle and Pod-Strut Geometry on Propulsive Performance of Pusher-Propeller Configurations, A thesis submitted to the School of Graduate Studies in partial fulfillment of the requirements for the degree of Master of Engineering, Faculty of Engineering and Applied Science Memorial University of Newfoundland.
- Podded Propulsor Tests and Extrapolation(2002). ITTC-Recommended Procedures, Proceedings of the 23rd ITTC.
- THE PROPULSION COMMITTEE, Final Report and Recommendations to the 22nd ITTC(1999). Proceeding of 22nd ITTC.
- The Specialist Committee on Azimuthing Podded Propulsion, Final Report and Recommendations to the 24th ITTC(2005), Proceedings of the 24th ITTC - Volume II.
- Tomi Veikonheimo(2006), Solving problems of propeller model scaling, The Naval Architect July/August, pp.51~53.
- Vladimir I. Krasilnikov, Alexander S. Achkinadze, Dmitry V. Ponkratov, Jia Ying Sun(2007), Prediction of Open Water Characteristics of Podded Propulsion Using a Coupled Viscous/Potential Solver, Ninth International Conference on Fast Sea Transportation FAST2007, p.311~321, Shanghai, China.
- Vladimir I. Krasilnikov, Aage Berg, Alexander S.Achkinadze, Dmitriy V.Ponkratov(2005), Numerical Study on Interaction Effects in The Pod Propulsive System, Proceedings of IWSH'2005, Shanghai China.